2nd International Seminar On "Utilization of Non-Conventional Energy Sources for Sustainable Development of Rural Areas
ISNCESR'16
17th & 18th March 2016

CFD Analysis of Vortex Tube Employing Different Length of the Tube

Dewashish Patel[1], Mayank Thakur[2], Jitendra kurre[3]

[1]CSVTU Bhili (C.G.), Shri Shankaracharya Group of Institutions, Junwani Bhilai (C.G.)
dev25689@gmail.com

[2] CSVTU Bhilai (C.G.), Shri Shankaracharya Group of Institutions, Junwani Bhilai (C.G.)
thakurmayank23@gmail.com

[3] CSVTU Bhilai (C.G.), Shri Shankaracharya Group of Institutions, Junwani Bhilai (C.G.)
jitendrakurre18@gmail.com

Abstract: ***Vortex tube is a simple device that produces hot and cold flow from a high pressure gas which is flowing tangentially to the inlet. The CFD model of present research is based on the analysis of Skye's experimental vortex tube. The vortex tube which is used in this model is equipped with a circular straight vortex tube with 3 inlets having a laminar flow of fluids. Since more than 3 inlets may produce high rotating flow inside the vortex tube and makes a complex compressible turbulent flow. The study is calculated by ANSYS 14.0 FLUENT software. The performance of vortex tube is studied for four different geometric models, which shows four different performances. In this article different length of 92 mm, 109 mm, 120 mm and 250 mm has been taken to examine the effect of length at the cold and hot outlet. In all different length 109 mm performs the best result.***

Keywords: Vortex Tube, cold orifice, hot orifice.

1. Introduction

Vortex tube is a simple device that produces hot and cold flow from a high pressure gas which is flowing tangentially to the inlet. It consisting of one or more inlet nozzle, vortex chamber, hot end tube, cold end tube, cold orifice and hot-end conical shape control valve. In a vortex tube, a tangentially flow of pressurized gas (generally air) by using one or more nozzle. Due to tangentially entry of pressurized fluid a circulating movement at high speed takes place inside the vortex tube. As the fluid flow through the vortex tube in circular motion, it splits into two low-pressure streams, one cold and the other hot. For a given inlet pressure and temperature, the temperature at the outlet may vary by varying the fraction of hot and cold streams with the help of a valve.

The Ranque-Hilsch vortex tube separates a compressed gas stream into two lower pressure streams with one stream having superior temperature and the other having inferior temperature than the inlet stream. This phenomenon is referred to as energy (temperature) separation. The tangential region of the flow is found to be warmer than the inlet gas, while the flow near the core region becomes colder than the inlet gas. The warm peripheral flow comes out through the annular space between the tube wall and the conical valve at the hot outlet. The colder core flow in the opposite direction comes out through the central orifice of the cold outlet.

Vortex tube is use in many industrial applications such as separating of gases, cooling of electronic component, dehumidification of air, cooling of environmental chambers, cooling and heating of equipment in laboratories dealing with explosive chemicals, cooling of fireman's suits, nuclear reactors etc.

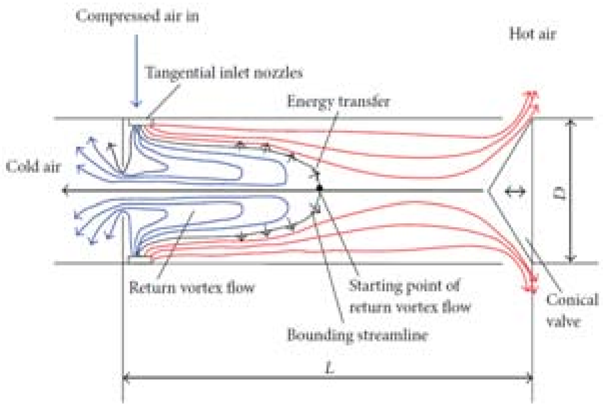


Figure 1 Flow pattern in Vortex Tube

Due to its simple construction with non-moving parts, compactness, light weight, reliability, low cost, low maintenance cost, safety, durability, instantaneous cooling and heating, ease in controlling output temperature, environmental friendly it is most widely used all over the globe. Having all this benefit it has a main limiting factor of low thermal efficiency restrict its uses in many application. Also the noise and availability of compressed gas restrict the use of vortex tube.

1.3 Objective of present study

The conclusion of the present study consists of a diminutive memorandum on various considerations which concern the performance and efficiency of vortex tube. The purpose of this research was to create a CFD model of a commercial vortex tube for use as a design tool in optimizing vortex tube performance. In the present study a simulation model is created to predict the affect of various parameters which involve to the performance of vortex tube. The simulation model is made by skye et. at. experimental model. The model is developed using three-dimension (3D) steady axi-symmetric.

2. Design of Simulation Model

This article gives a direction to study the flow of fluid in vortex tube by using least squares cell based method and creating simulation model of vortex tube by the help of ANSYS 14.0 FLUENT model. In the new regarding the Skye's vortex tube is modelled numerically with 3 circular straight inlets such that the each nozzle areas are kept constant to all sets. This is due to fact that this article believes that circular straight nozzles can play very considerable role in appropriately operating of a vortex tube even for a few number of nozzles.

The assumptions for computation of the vortex tube flows were made as follows. It consists of rotational pressurized gas inlets and two axial orifices for cold and hot stream with air as a working fluid. Since the chamber consists of 3 inlet nozzles, the CFD models are assumed to be rotational flow. The Dimensional geometric information of these vortex tube models is presented in table 2.1

Table 2.1: Geometric summery of CFD models taken from skye et. al. [1]

Measurements	Present vortex tube
Working tube length	106 mm
Working tube internal diameter	11.4 mm
Nozzle height	0.97 mm
Nozzle width	2.82 mm
Nozzle total area	8.2 mm^2
Cold exit diameter	6.2 mm
Hot exit area	95 mm^2

3. Governing equation of Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics which uses numerical methods and algorithms to solve and analyse problems which involving fluid flow, heat transfer, combustion etc. The governing equations (i.e. conservation of mass, conservation of momentum and conservation of energy) of fluid motion are usually described through fundamental mathematical equations which uses partial differential equations (PDE).

A general outline of partial differential equation which can signify various conservation laws of fluid flow is given below

Continuity equation

$$\frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} = 0 \qquad \text{Eq. (1)}$$

X-Momentum equation

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} \right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \qquad \text{Eq. (2)}$$

Y-Momentum equation

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} \right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \qquad \text{Eq. (3)}$$

Energy equation

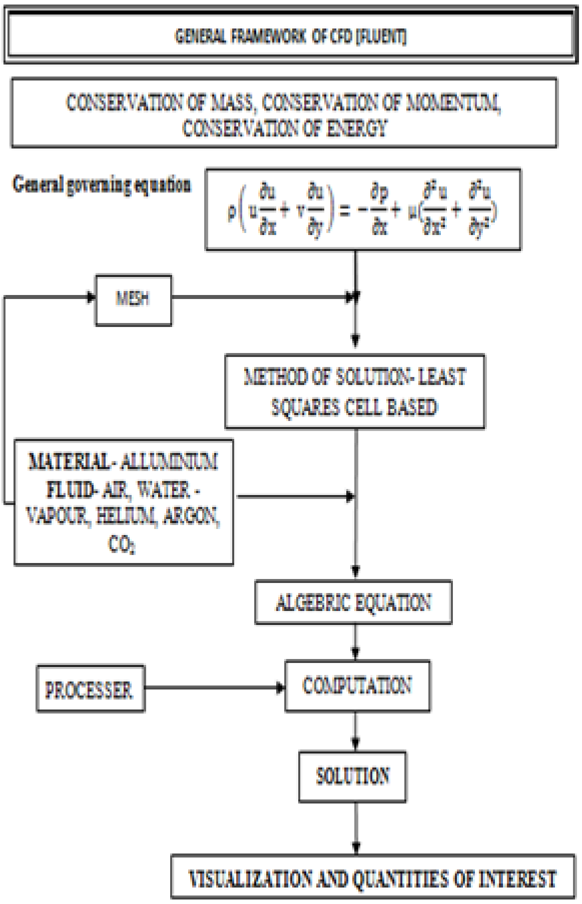
$$\rho C_p\left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} \right) = K\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \qquad \text{Eq. (4)}$$

4. CFD analysis

The ANSYS 14.0 FLUENT Workbench interface is arranged into two primary areas: The Toolbox and the Project Schematic. The Toolbox contains the system templates that you can use to build a project. The Project Schematic is the area of the interface where you will manage your project. In addition, you will see a toolbar and a menu bar with frequently used functions. The blog diagram which shows the primary features is shown below.

A steady three dimensional model is prepared. A pressure based implicit scheme is used for analysis. Standard Skye's model with viscous dissipation term enabled is employed. Air (behaves as ideal gas) is used as working substance. Simple algorithm is used for the solution of equation for compressible flow.

Table 4.1: Blog diagram of CFD analysis



5. Validation

In this attention is focused on validation of simulation model of vortex tube made by CFD 14.0 FLUENT. The temperature distribution is studied with the help of stream function and contours of temperature respectively. The vortex tube has a huge effect on the performance by changing its major length. In this study a mean value of length is calculated for which it has a higher performance for every type of fluids.

5.1 **Mesh component system**

The Mesh cell in Fluid Flow analysis systems or the Mesh component system is used to produce a mesh using the Meshing application. It can also be used to import an existing mesh file. When the geometry is imported it must be specified to a definite nodes and elements. The meshing of the geometric model of vortex tube contains 9298 nodes, and 45035 elements. Below Fig. 5.1, Fig. 5.2 and Fig 5.3 shows the meshing of vortex tube in isometric view, top view and side view respectively.

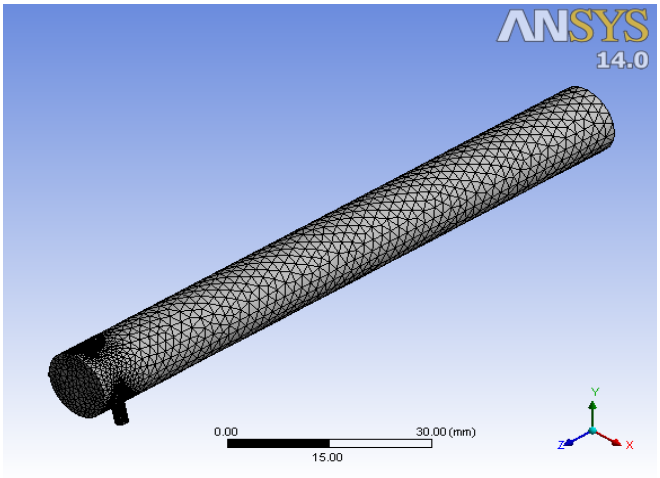


Figure 5.1 Isometric view of Vortex Tube

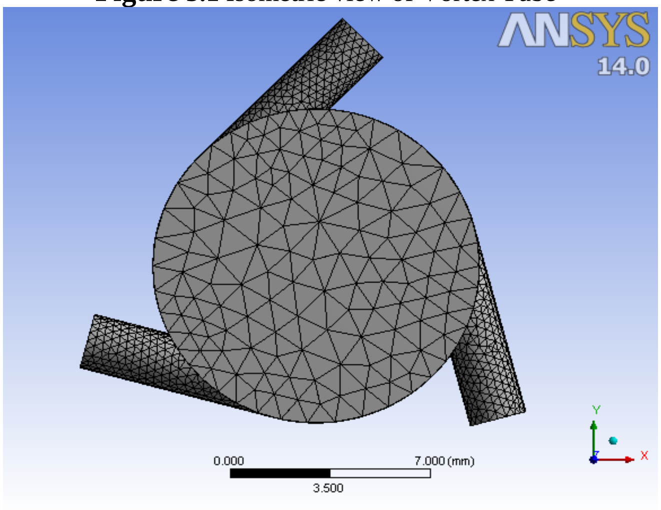


Figure 5.2 Top view of Vortex Tube

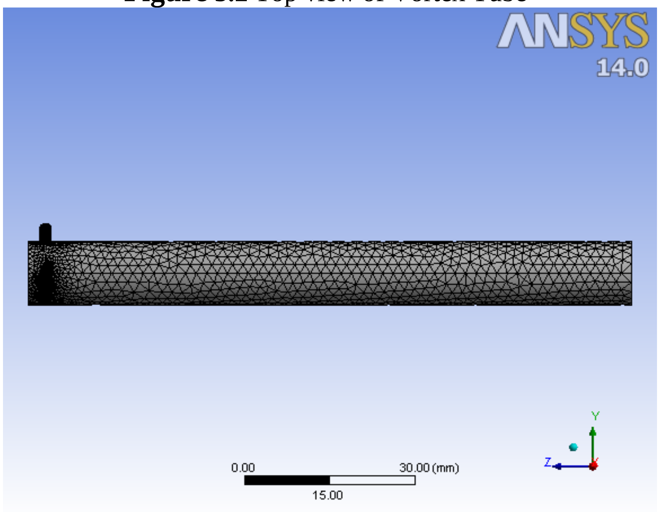


Figure 5.3 Front view of Vortex tube

5.2 Comparison of length of Vortex Tube

The other most important investigation is done on the effect of the length of vortex tube. By the investigation it has been found that the effect of the length of vortex tube plays an extensive role on the effect of temperature at the cold and hot outlet. The performance of vortex tube is studied four different geometric models, which shows four different performance In this article different length of 92 mm, 109 mm, 120 mm and 250 mm has been taken to examine the effect at the cold and hot outlet. When the length of vortex tube is changed the performance of vortex tube also change. A systematic table 5.2 has been drawn below to show the effect of temperature at the cold and hot outlet by taking different length of tube.

Table 5.1 Effect of length on performance of Vortex Tube

Sr. No.	Length of Vortex Tube	Cold Outlet Temp	Hot Outlet Temp	Temp Difference ∆ T
1.	**92 mm**	251.2332	287.3759	36.1427
2.	**109 mm**	249.5444	298.4832	51.2684
3.	**120 mm**	250.2683	296.8324	46.5641
4.	**250 mm**	253.2735	288.5893	35.3158

6. Conclusion

The three-dimension steady axi-symmetric model is use to predict the effect of various length of vortex tube. The length of vortex tube has immense effect on the performance of vortex tube. The researchers who studying vortex tube, have suggested different values for length, but should be said that the length, can be different for each definite project .In the present study the tube with L = 109 mm presented the finest results for the highest possible temperatures.

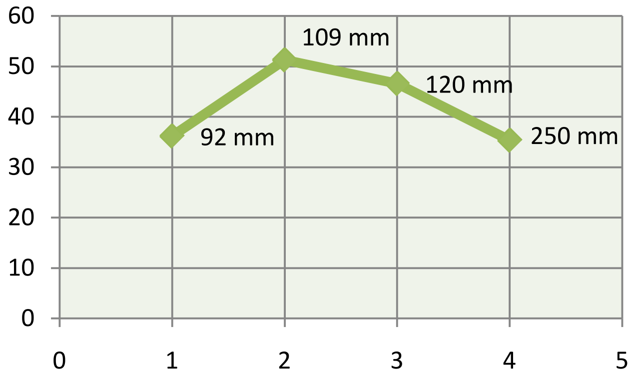


Figure 6.1 Comparisons between different lengths.

By this model following conclusion are obtained.

a) A simulation model is created to study the effect of different parameter of the vortex tube.
b) In second consideration when the length of vortex tube is improved to 109 mm it shows the difference of hot and cold outlet as 51.2684 K.

References

[1] H. M. Skye, G. F. Nellis, S.A. Klein, 2009 "Comparison of CFD Analysis to Empirical Data in a Commercial Vortex Tube" WI 53706.

[2] Pongjet Promvonge and Smith Eiamsa-ard, 2005 "Investigation on the Vortex Thermal Separation in a Vortex Tube Refrigerator" ScienceAsia 31: 215-223.
[3] Smith Eiamsa-ard, Pongjet Promvonge, 2006 "Numerical investigation of the thermal separation in a Ranque–Hilsch vortex tube", Heat and Mass Transfer 50 (2007) 821–832.
[4] http://www.arizonavortex/vortextubes.htm
[5] http://www.newmanstool.com/votex.com
[6] Nader Pourmahmoud & Abdol Reza Bramo, " 2011, " The effect of l/d ratio on the temperature sepration in the counter flow vortex tube", IJRRAS_6_1_07.
[7] Nader Pourmahmoud, Amir Hassan Zadeh, Omid Moutaby, and Abdolreza Bramo, 2012 "computational fluid dynamics analysis of helical nozzles effects on the energy separation in a vortex tube" Vol. 16, No. 1, pp. 151-166.
[8] Jayakumar D. Golhar, B.R. Rathode, A.N. Pawar, 2012, "Experimental Investigation and Optimization of Vortex Tube with Regard to Nozzle Diameter" International Conference on Benchmarks in Engineering Science and Technology ICBEST.
[9] Rahul B Patel, Dr. V.N. Bartatia, Tausif M Shaikh, 2012, "A Review On Exprimental & Numerical Investigation Of Ranque Hilsch Vortextube", IJERA ISSN: 2248-9622.
[10] T. Dutta, K. P. Sinhamahapatra, and S. S. Bandyopadhyay, 2013, "CFD Analysis of Energy Separation in Ranque-Hilsch Vortex Tube at Cryogenic Temperature", Article ID 562027.
[11] H.R. Thakare, Y.R. Patil and A.D. Parekh, 2013, "A Review of Computational Studies of Temperature Separation Mechanism in Vortex Tube", Bonfring International Journal of Industrial Engineering and Management Science, Vol. 3